Applications of Computational Fluid Dynamics in Mechanical Engineering – Series 4

Editors:
Nofrizalidris Darlis
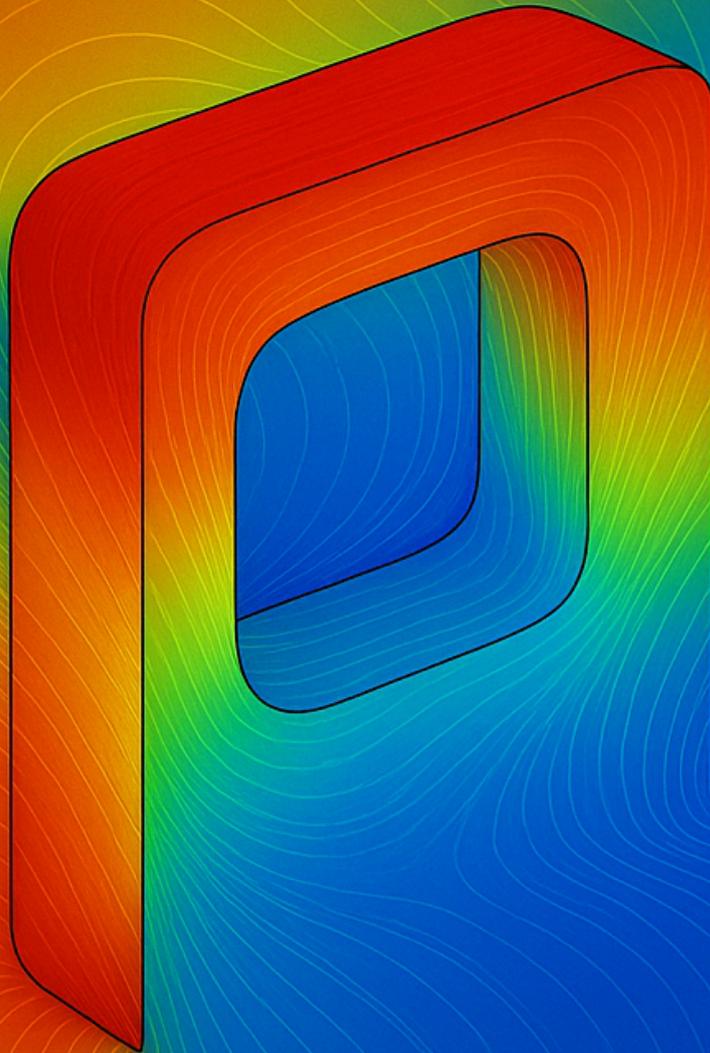Muhammad Noor Afiq Witri Muhammad Yazid
Nurulnatisya binti Ahmad

Abstract: Computational Fluid Dynamics (CFD) has become an indispensable tool in modern mechanical engineering, enabling detailed analysis of fluid flow, heat transfer, and thermal performance across a wide range of systems. This series continues to highlight the diverse studies contributed by researchers from different parts of the world, including distinguished authors from Nigeria and Indonesia, reflecting the global scope and collaboration in CFD. The topics covered include the thermal analysis and temperature distribution in an earth pipe air heat exchanger, the heat transfer behaviour in an infant warmer, and the exergy analysis of a combustor, each emphasizing how CFD enhances understanding and design efficiency. Further sections explore the CFD analysis of a shell and double concentric tube heat exchanger, heat transfer enhancement in microchannel heat sinks using mxene-based nanofluids, and the thermal performance evaluation of fuel gas preheaters. Together, these studies illustrate how CFD serves as a powerful tool for innovation, optimization, and sustainable development in heat transfer applications.

Keywords: Computational Fluid Dynamics, mechanical engineering, heat transfer

Applications of Computational Fluid Dynamics Simulation in Mechanical Engineering

EDITORS:
NOFRIZALIDRIS DARLIS
NURULNATISYA AHMAD
MUHAMMAD NOOR AFIQ WITRI
MUHAMMAD YAZID

Applications of Computational Fluid Dynamics in Mechanical Engineering – Series 4

Applications of Computational Fluid Dynamics in Mechanical Engineering –
Series 4

EDITORS:
NOFRIZALIDRIS DARLIS
MUHAMMAD NOOR AFIQ WITRI MUHAMMAD YAZID
NURULNATISYA BINTI AHMAD

Editor:
Nofrizalidris Darlis, Muhammad Noor Afiq Witri Muhammad Yazid, Nurulnatisya Ahmad

CONTENTS

PREFACE

Computational Fluid Dynamics (CFD) has become an indispensable tool in modern mechanical engineering, enabling detailed analysis of fluid flow, heat transfer, and thermal performance across a wide range of systems. This series continues to highlight the diverse studies contributed by researchers from different parts of the world, including distinguished authors from Nigeria and Indonesia, reflecting the global scope and collaboration in CFD. The topics covered include the thermal analysis and temperature distribution in an earth pipe air heat exchanger, the heat transfer behaviour in an infant warmer, and the exergy analysis of a combustor, each emphasising how CFD enhances understanding and design efficiency. Further sections explore the CFD analysis of a shell-and-double concentric tube heat exchanger, heat transfer enhancement in microchannel heat sinks using Mxene-based nanofluids, and the thermal performance evaluation of fuel gas preheaters. Together, these studies illustrate how CFD serves as a powerful tool for innovation, optimisation, and sustainable development in heat transfer applications.

List of Editors

NOFRIZALIDRIS DARLIS
MUHAMMAD NOOR AFIQ WITRI MUHAMMAD YAZID
NURULNATISYA AHMAD

CHAPTER 1

THERMAL ANALYSIS AND TEMPERATURE DISTRIBUTION IN EARTH PIPE AIR HEAT EXCHANGER

Kumaran Kalimuthu[1], Muhammad Noor Afiq Witri Muhammad Yazid[1], Aminuddin Saat[1], Fazila Mohd Zawawi[1], Asral[2], Nofrizalidris Darlis[3]

[1]Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 Johor Bahru, Johor, Malaysia

[2]Department of Mechanical Engineering, Universitas Riau, Indonesia

[3]Centre of Automotive & Powertrain Technology, Faculty of Engineering Technology, University Tun Hussein Onn Malaysia, Edu Hub Pagoh, 84600, Muar, Johor, Malaysia

ABSTRACT

Temperature of air that flows into underground tunnel can be cooled down, which can be utilized for reducing the cooling load of buildings. The present chapter aims to analyse the outlet air temperature of an earth pipe air heat exchanger (EPAHE) system. A transient and implicit model based on computational fluid dynamics was developed to predict the thermal performance of the EPAHE. Computational fluid dynamics (CFD) simulation is employed to determine the temperature distribution across the EPAHE system for different types of soil and with different range of air velocities. The transient study of temperature profile, velocity profile of the piping model has been conducted by using Renormalized Group (RNG) $k-\varepsilon$ turbulent model. Effects of the operating parameters such as the type of soil and air velocity on the thermal performance of EPAHE systems are discussed. The 23.42 m long and 2.75 m depth inside the soil of EPAHE system discussed in this paper gives cooling in the range of 10.0 to 13.0°C for the flow velocities 1 to 5 m/s in four different types of soils. We found the type of soil buried, and the flow of air velocity contributed to the major effect of the performance of EPAHE system.

Keywords: earth pipe air heat exchanger, CFD, thermal analysis

CHAPTER 2

ANALYSIS OF HEAT TRANSFER OF INFANT WARMER

Mohammad Hatta Ramlee[1], Ab Aziz Mohd Yusof [2],
Kamariah Md Isa[2], Nurulnatisya Ahmad[2], Muhammadu Masin Muhammadu[3],
Nofrizalidris Bin Darlis[4]

[1] Universiti Teknologi Malaysia, Skudai, Johor, Malaysia.

[2] Faculty of Mechanical Engineering, Universiti Teknologi MARA, UiTM, Johor
Branch, Malaysia.

[3]Department: Mechanical engineering, Federal University of Technology, Minna,
Nigeria.

[4]Centre of Automotive & Powertrain Technology, Faculty of Engineering
Technology, University Tun Hussein Onn Malaysia,
Edu Hub Pagoh, 84600, Muar, Johor, Malaysia

ABSTRACT

An infant warmer is an overhead heating device used to help infants regulate their body heat to achieve their normal body temperature. However, problems may arise in obtaining the correct heat settings. The objective of this study is to determine the most appropriate operating conditions of such units. A combined analysis of heat transfer from the heat source and the influence of the surrounding environment temperature is performed to calculate the temperature distribution for the infant warmer. A simulation application is used to recreate the design and run the simulations according to the boundary conditions as a guideline. The results for a single heat source show a very non-uniform temperature distribution. On the other hand, the results for the double heat source show a very uniform heat temperature distribution. Based on the observations, it is recommended that a dual heat source be used for an infant warmer for more favourable results.

Keywords: Infant warmer, Heat transfer, CFD

CHAPTER 3

EXERGY ANALYSIS OF A COMBUSTOR

Amirul Affannuhafiz Abdul Muttaleb[1], Mohd Rosdzimin Abdul Rahman[1], Mohd Rashdan Saad[1], Mohd Faizal Hasan[2], Kahar Osman[2], Ishkrizat Taib[3]

[1]Jabatan Kejuruteraan Mekanikal, Fakulti Kejuruteraan,
Universiti Pertahanan Nasional Malaysia,
Kem Sg Besi, 57000, Kuala Lumpur, Malaysia

[2]Faculty of Mechanical Engineering, Universiti Teknologi Malaysia,
81310 Johor Bahru, Johor, Malaysia

[3]Fakulti Kejuruteraan Mekanikal dan Pembuatan,
Universiti Tun Hussein Onn Malaysia,
86400, Parit Raja, Johor, Malaysia

ABSTRACT

This study presents an exergy analysis of a premixed hydrogen/air miniature combustor, focusing on the impact of different bluff body ratios on combustion characteristics and exergy performance. The primary objective was to investigate the combustion efficiency and stability at varying bluff body geometries and inlet velocities, with an emphasis on improving energy conversion. The results show that the bluff body ratio of 0.3 provided the best overall combustion performance, with stable flame development and efficient energy conversion at inlet velocities between 10 m/s and 12 m/s. Higher velocities, specifically at 14 m/s, led to decreased flame stability and reduced exergy efficiency due to poor mixing and potential cooling effects. The exergy efficiency peaked at velocities around 10 m/s, demonstrating the system's ability to effectively convert energy into useful work while minimizing energy loss. This analysis contributes to the development of more efficient combustion systems, offering insights into optimal design parameters for sustainable energy use. The findings support the achievement of the United Nations Sustainable Development Goal (SDG) 7, "Affordable and Clean Energy," by advancing the design of miniature combustors with improved energy efficiency.

CHAPTER 4

COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF A SHELL-AND-DOUBLE CONCENTRIC TUBE HEAT EXCHANGER

S.M. Shahril[1], Kahar Oman[2], M.K.R Hashim[1], A.M.N. Bariah[3], Ummikalsom Abidin[2], Nofrizalidris Darlis[4]

[1]Faculty of Mechanical Engineering Technology, University Malaysia Perlis (UniMAP), Pauh Putra, 02600 Arau, Perlis, Malaysia.

[2]Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 Johor Bahru, Johor, Malaysia

[3]Faculty of Electrical Engineering Technology, Universiti Malaysia Perlis(UniMAP), Pauh Putra Campus, 02600 Arau, Perlis, Malaysia

[4]Centre of Automotive & Powertrain Technology, Faculty of Engineering Technology, University Tun Hussein Onn Malaysia, Edu Hub Pagoh, 84600, Muar, Johor, Malaysia

ABSTRACT

Shell-and-tube heat exchangers (STHEX) have been used for several decades. Conventionally, to increase the thermos-hydraulic performance of classical heat exchangers, the overall length of tubes has to be increased. This contributes to major disadvantages in terms of classical heat exchangers' design, particularly considering the economic aspect. In this study, the thermo-hydraulic performance analysis of a shell-and-double concentric tube heat exchanger (SDCTHEX) is carried out using commercially available Computational Fluid Dynamics (CFD) software ANSYS FLUENT. The results show that the average percentage increase in overall heat transfer rate per overall pressure drop of SDCTHEX, with inner tube diameter equal to 8/12 mm/mm, is nearly 343% higher than that of STHEX, while the total friction power expenditure of SDCTHEX is reduced by around 85.5% as compared to that of STHEX. Also, the overall heat transfer rate per overall pressure drop of SDCTHEX is sensitive to inner tube diameter. It is found that U=DP for the mass flow rate of 22.5 kg/s is maximum and found to be about 400% higher at inner tube diameter of 12/16 mm/mm with respect to the STHEX. The

CHAPTER 5

HEAT TRANSFER ENHANCEMENT IN MICROCHANNEL HEAT SINKS USING MXENE-BASED NANOFLUIDS: A CFD APPROACH

Mohamad Nur Hidayat Mat[1], Nurizzatul Atikha Rahmat[1], Muhamad Mat Noor[1], Eliza M. Yusup[2], Muhammad Noor Afiq Muhammad Yazid[3], Nofrizalidris Darlis[4]

[1]Faculty of Mechanical and Automotive Engineering Technology, Universiti Malaysia Pahang Al-Sultan Abdullah, 26600 Pekan, Pahang, Malaysia

[2]Faculty of Mechanical & Manufacturing Engineering, Universiti Tun Hussein Onn, Parit Raja, 86400, Malaysia

[3]Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310, Johor Bahru, Johor, Malaysia

[4]Centre of Automotive & Powertrain Technology, Faculty of Engineering Technology, University Tun Hussein Onn Malaysia, Edu Hub Pagoh, 84600, Muar, Johor, Malaysia

ABSTRACT

As advancements in electronic technologies continue, the need for effective thermal management becomes increasingly vital. This research investigates the application of MXene-enhanced nanofluids at different volume concentrations in microchannel heat sinks to enhance cooling efficiency in electronic devices. Using numerical simulations based on a Eulerian multiphase framework, the study analyses the behavior and thermal properties of these nanofluids. Volume fractions ranging from 0.01% to 0.04% were assessed, and simulation results were cross validated with experimental data to ensure accuracy. At a Reynolds number of 300, a nanofluid concentration of 0.04 vol.% demonstrated a 20.1% decrease in thermal resistance when compared to water. Meanwhile, at Re 1000, the heat transfer coefficient improved by 29.4%. These outcomes indicate the promising potential of MXene-based nanofluids as advanced coolants for electronic heat sinks. The insights from this study are relevant across various electronic applications, such as power electronics, LEDs, and integrated circuits,

CHAPTER 6

THERMAL PERFORMANCE ANALYSIS OF FUEL GAS PREHEATERS USING CFD TECHNIQUES

Thivaakar Rajaratinam[1], Kamariah Md Isa[2], Nur Hazwani Mokhtar[3], Ab Aziz Mohd Yusof[2], Aminuddin Saat[1], Kahar Osman[1], Nofrizalidris Darlis[4]

[1]Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310, Johor Bahru, Johor, Malaysia

[2]Faculty of Mechanical Engineering, College of Engineering, Universiti Teknologi MARA, UiTM, Johor Branch, Masai, Malaysia

[3] Fakulti Teknologi dan Kejuruteraan Mekanikal, Universiti Teknologi Melaka, Melaka, Malaysia

[4]Centre of Automotive & Powertrain Technology, Faculty of Engineering Technology, University Tun Hussein Onn Malaysia, Edu Hub Pagoh, 84600, Muar, Johor, Malaysia

ABSTRACT

Gas turbines have been a cornerstone of power generation technology since their introduction in 1939, operating through the continuous interaction of air compression, fuel combustion, and turbine expansion. Modern designs employ axial compressors equipped with inlet guide vanes (IGV) and variable guide vanes (VGV) to enhance airflow control and efficiency. This study focuses on the optimisation of fluid velocities within a fuel gas preheater, an essential component in gas turbine systems. Using computational fluid dynamics (CFD) analysis, various flow conditions were evaluated to determine their impact on thermal performance. The results revealed that setting the natural gas velocity at 5 m/s and the water velocity at 1 m/s produced the most favourable outcomes, yielding the highest temperature rise, superior heat transfer rates, and a notable pressure drop on the shell side, indicative of enhanced energy exchange. Among the CFD models tested, the k–ε turbulence model coupled with a pressure-based solver demonstrated the most accurate and physically consistent predictions. These findings underscore the importance of optimised flow dynamics in improving heat exchanger efficiency and provide valuable insights for advancing gas turbine

Applications of Computational Fluid Dynamics Simulation in Mechanical Engineering

Computational Fluid Dynamics (CFD) has become an indispensable tool in modern mechanical engineering, enabling detailed analysis of fluid flow, heat transfer, and thermal performance across a wide range of systems. This series continues to highlight the diverse studies contributed by researchers from different parts of the world, including distinguished authors from Nigeria and Indonesia, reflecting the global scope and collaboration in CFD. The topics covered include the thermal analysis and temperature distribution in an earth pipe air heat exchanger, the heat transfer behaviour in an infant warmer, and the exergy analysis of a combustor, each emphasising how CFD enhances understanding and design efficiency. Further sections explore the CFD analysis of a shell-and-double concentric tube heat exchanger, heat transfer enhancement in microchannel heat sinks using Mxene-based nanofluids, and the thermal performance evaluation of fuel gas preheaters. Together, these studies illustrate how CFD serves as a powerful tool for innovation, optimisation, and sustainable development in heat transfer applications.

Penerbit
UTHM